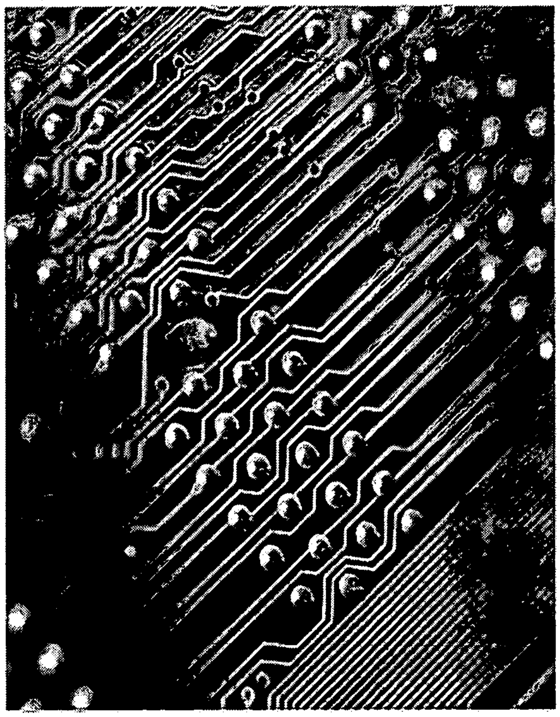
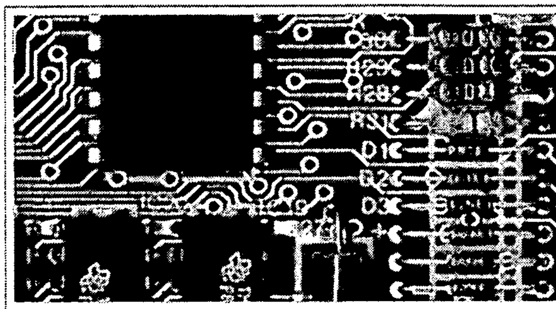
Printed circuit board

From Wikipedia, the free encyclopedia



Close-up photo of one side of a motherboard PCB, showing conductive traces, vias and solder points for through-hole components on the opposite side.



ZX Spectrum PCB, showing conductive traces, through-hole pathes into the opposite side and element montage

A **printed circuit board** or **PCB** mechanically supports and electrically connects electronic components using conductive pathways, or *traces*, etched from copper sheets laminated onto a non-conductive *substrate*. Alternative names are **printed wiring board** or **PWB** or **etched wiring board**

PCBs are rugged, inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either wire-wrapped or point-to-point constructed circuits, but are much cheaper, faster, and consistent in high volume production.

Contents

- 1 History
- 2 Physical composition
  - 2.1 Substrates
- 3 Design
  - 3.1 Electronic Design Automation
- 4 Manufacturing
  - 4.1 Patterning
  - 4.2 Lamination
  - 4.3 Drilling
  - 4.4 Solder plating and solder resist
  - 4.5 Silk screen

  - 4.6 Populating
  - 4.7 Protection and packaging
- 5 'Cordwood' construction
- 6 Surface-mount technology
- 7 See also
- 8 References
- 9 External links

History

The inventor of the printed circuit was probably the Austrian engineer Paul Eisler (1907–1995) who, while working in England, made one circa 1936 as part of a radio set. Around 1943 the USA began to use the technology on a large scale to make rugged radios for use in World War II. After the war, in 1948, the USA released the invention for commercial use. Printed circuits did not become commonplace in consumer electronics until the mid-1950s.

Before printed circuits, point-to-point construction was used. For prototypes, or small production runs, wire wrap can be more efficient.

Originally, every electronic component had wire leads, and the PCB had holes drilled for each wire of each component. The components' leads were then passed through the holes and soldered to the PCB trace. This method of assembly is called *through-hole* construction. Soldering could be done automatically by passing the board over a ripple, or wave, of molten solder in a wave-soldering machine. Through-hole mounting is still used.

However, the wires and holes are wasteful. It costs money to drill the holes, and the protruding wires are merely cut off.

Instead of using through-hole parts, often 'surface mount' parts are used instead. See *Surface-mount technology* below.

Physical composition

Most PCBs are composed of between one and sixteen conductive layers separated and supported by layers of insulating material (*substrates*) laminated (glued) together.

Layers may be connected together through drilled holes called vias. Either the holes are electroplated or small rivets are inserted. High-density PCBs may have *blind vias*, which are visible only on one surface, or *buried vias*, which are visible on neither.

Substrates

Low-end consumer grade PCB substrates frequently are made of paper impregnated with phenolic resin, sometimes branded "Pertinax". They carry designations such as XXXP, XXXPC, and FR-2. The material is inexpensive, easy to machine by drilling, shearing and cold punching, and causes less tool wear than glass fiber reinforced substrates. The letters "FR" in the designation indicate Flame Resistance.

High-end consumer and industrial circuit board substrates are typically made of a material designated FR-4. This consists of a woven fiberglass mat impregnated with a flame resistant epoxy resin. It can be drilled, punched and sheared, but due to its abrasive glass content requires tools made of tungsten carbide for high volume production.

Due to the fiberglass reinforcement, it exhibits about five times higher flexural strength and resistance to cracking than paper-phenolic types, albeit at higher cost.

PCBs for high power radio frequency (RF) work use plastics with low dielectric constant (permittivity) and dissipation factor, such as Rogers® 4000, Rogers® Duroid, DuPont® Teflon® (types GT and GX), polyimide, polystyrene and cross-linked polystyrene. They typically have poorer mechanical properties, but this is considered an acceptable engineering tradeoff in view of their superior electrical performance.

PCBs designed for use in vacuum or in zero gravity, as in spacecraft, being unable to rely on convection cooling, often have thick copper or aluminum cores to dissipate heat from electrical components.

Not all circuit boards use rigid core materials. Some are designed to be completely or partially flexible, using DuPont's® Kapton® polyimide film, and others. This class of boards, sometimes called *flex circuits*, or *rigid-flex circuits*, respectively, are difficult to create but have many applications. Sometimes they are flexible to save space (PCBs inside cameras and hearing aids are almost always made of flex circuits so they can be folded up to fit into the limited available space). Sometimes, the flexible part of the circuit board is actually being used as a cable or moving connection to another board or device. One example of the latter application is the cable connected to the carriage in an inkjet printer.

Design

Usually an electronics or electrical engineer designs the circuit, and a layout specialist designs the PCB. The designer must obey numerous PCB layout guidelines to design a PCB that functions correctly, yet is inexpensive to manufacture.

Electronic Design Automation

PCB designers often use electronic design automation (EDA) to produce a *layout*. The EDA program stores design information, facilitates editing the design, and can also automate repetetive design tasks.

The first stage is converting the circuit schematic into a *net list*. The net list is conceptually a list of component pins and the circuit nodes, or *nets*, that each pin connects to. Often the *schematic capture* EDA program, operated by a circuit design engineer, is responsible for *netlist generation*, and the netlist is *imported* into the PCB layout program.

The next step is to decide the position of each device. The easy way to do this is to specify a grid of lettered rows and numbered columns where the devices should go. The computer then assigns pin 1 of each device in the *bill of materials* to a grid location. Typically, the operator may assist the automated placement routine by specifying *rooms*, or specific regions of the board, where certain groups of components should be placed. For example, the parts associated with a power supply subcircuit might be assigned to a region near the power input connector. In other cases devices may be manually placed, either to optimize the electrical performance of the circuit, or to place components such as knobs, switches, and connectors as required by the mechanical design of the system.

The computer then *explodes* the device list into a complete pin list for the board by using templates from a *library* of *footprints* associated with each type of device. Each footprint is a map of a device's pins, usually with a recommended pad and drill hole layout for each device. The library allows the footprint to be drawn only once, and then shared by all devices of that type.

In some systems, high-current pads are identified in the device library, and the associated nets are flagged for attention by the pcb designer. High current runs require wider traces, and the designer or circuit design engineer

usually decides the width.

The computer program then merges the netlist (sorted by pin name) with the pin list (sorted by pin name), transferring the physical coordinates of the pin list to the netlist. The netlist is then resorted, by net name.

Some systems can optimize the design by swapping the positions of parts and logic gates to reduce the length of copper runs. Some systems also automatically discover power pins in the devices, and generate runs or vias to the nearest power plane or conductor.

The programs then try to route each net in the signal-pin list, finding some sequence of connections in the available layers. Often layers are assigned to power and ground, with one layer to vertical, and another to horizontal wires. The power layers shield the circuits from noise.

The routing problem is equivalent to the travelling salesman problem, and is therefore NP complete, and therefore not amenable to a perfect solution. One practical routing algorithm is to pick the pin farthest from the center of the board, then use a greedy algorithm to select the next-nearest pin with the same signal name.

After automated routing, usually there is a list of nets that must be manually routed.

Once routed, the system may have a series of strategy subroutines to reduce the production cost of the PCB. For example, one routine might remove unneeded vias (each via is a drill hole, and costs money to make). Another might round edges of conductor runs, and widen or move runs apart to maintain safe spacings. Another strategy might adjust large copper areas so that they form nets, or large blank areas may get unconnected "checks" of copper. The nets and checks reduce pollution by extending the life of the etchant batch, and speed production by evening-out the copper concentration in the etching bath.

Some systems provide *design rule checking* to validate the design for electrical interference, excessive resistance, heat flow and other common errors.

The silk-screen, solder mask, and sometimes conformal coat are often designed as auxiliary layers.

Finally, the copper layers are then converted to gerber files, a format of numerical control file for a photoplotter. Historically, an additional *aperture file* was required to link each numerically designated aperture referred to in the gerber file with an actual shape to be plotted. Newer gerber files embed the aperture information in the gerber file itself. The hole locations are encoded in *drill files*. The drill files may be sorted to minimize drill-head movement time, and bit changes.

Manufacturing

Patterning

The vast majority of 'printed circuit boards' are made by adhering a layer of copper over the entire substrate, sometimes on both sides, (creating a "blank PCB") then removing unwanted copper after applying a temporary mask (e.g. by etching in an acid), leaving only the desired copper traces. A few PCBs are made by *adding* traces to the bare substrate usually by a complex process of multiple electroplating. Some PCBs have trace layers inside the PCB and are called *multi-layer* PCBs. These are formed by bonding together separately etched thin boards. After the circuit board has been manufactured, components are connected to the traces by soldering them to the board.

There are three common methods used for the production of printed circuit boards:

1. **Silk screen printing** uses etch-resistant inks to protect the copper foil. Subsequent etching removes the unwanted copper. Alternatively, the ink may be conductive, printed on a blank (non-conductive) board. The latter technique is also used in the manufacture of hybrid circuits.
2. **Photoengraving** uses a photomask and chemical etching to remove the copper foil from the substrate. The photomask is usually prepared with a photoplotter from data produced by a technician using computer-aided PCB design software. Laser-printed transparencies are sometimes employed for low-resolution photoplots.[1] (http://www.fullnet.com/u/tomg/gooteepc.htm)
3. **PCB Milling** uses a 2 or 3 axis mechanical milling system to mill away the copper foil from the substrate. A PCB milling machine (referred to as a 'PCB Prototyper') operates in a similar way to a plotter, receiving commands from the host software that control the position of the milling head in the x, y, and (if relevant) z axis. Data to drive the Prototyper is extracted from files generated in PCB design software and stored in HPGL or Gerber file format.

Lamination

Some PCBs have trace layers inside the PCB and are called *multi-layer* PCBs. These are formed by bonding together separately etched thin boards.

Home | Login | Logout | Access Information | Alerts |

IEEE Xplore RELEASE 2.1

Welcome United States Patent and Trademark Office

☐ **Search Results**

BROWSE SEARCH IEEE XPLORE GUIDE

Results for **"(((density) and (variation or difference or variance) and (pcb or printed circuit board))<in>m..."** e-mail

Your search matched **18** of **1318251** documents.

A maximum of **100** results are displayed, **25** to a page, sorted by **Relevance** in **Descending** order.

» Search Options

View Session History

New Search

» Key

IEEE JNL	IEEE Journal or Magazine
IEE JNL	IEE Journal or Magazine
IEEE CNF	IEEE Conference Proceeding
IEE CNF	IEE Conference Proceeding
IEEE STD	IEEE Standard

Modify Search

(((density) and (variation or difference or variance) and (pcb or printed circuit board)) Search

Check to search only within this results set

Display Format: Citation Citation & Abstract

view selected items **Select All** **Deselect All**

1. **Microvias and flex-an enabling MCM-L technology**
Keating, J.; Larmouth, R.;
Multichip Modules and High Density Packaging, 1998. Proceedings. 1998 7th I Conference on
15-17 April 1998 Page(s):342 - 348
Digital Object Identifier 10.1109/ICMCM.1998.670805
AbstractPlus | Full Text: PDF(812 KB) **IEEE CNF**
Rights and Permissions

2. **Solder joint strain reduction of leadless IC packages through finite eleme tailored PCB construction**
Thompson, K.R.;
Electronic Manufacturing Technology Symposium, 1989, Proceedings. Japan I Sixth IEEE/CHMT International
26-28 April 1989 Page(s):105 - 111
Digital Object Identifier 10.1109/IEMTS.1989.76118
AbstractPlus | Full Text: PDF(408 KB) **IEEE CNF**
Rights and Permissions

3. **Effects of printed-circuit-board layout on power switch case-to-ambient t resistance**
Yang Chen; Guozhu Chen; Keyue Smedley;
Industrial Electronics Society, 2003. IECON '03. The 29th Annual Conference
Volume 1, 2-6 Nov. 2003 Page(s):694 - 698 vol.1
Digital Object Identifier 10.1109/IECON.2003.1280066
AbstractPlus | Full Text: PDF(355 KB) **IEEE CNF**
Rights and Permissions

4. **FDTD analysis on the current and electric field distribution in a PCB strip**
Yung-Kyu Kim;
Electromagnetic Compatibility, 2000. IEEE International Symposium on
Volume 2, 21-25 Aug. 2000 Page(s):691 - 694 vol.2
Digital Object Identifier 10.1109/ISEMC.2000.874704
AbstractPlus | Full Text: PDF(552 KB) **IEEE CNF**
Rights and Permissions

5. **Thermal Design Based on Surface Temperature Mapping**
de Jong, E.C.W.; Ferreira, J.A.; Bauer, P.;

Power Electronics Letters, IEEE
Volume 3, Issue 4, Dec. 2005 Page(s):125 - 129
Digital Object Identifier 10.1109/LPEL.2005.860625

AbstractPlus | Full Text: PDF(840 KB) IEEE JNL
Rights and Permissions

6. **Etching process analysis based on etchant flow for high-density build-up**
Noma, H.; Nakanishi, T.;
Electronics Packaging Technology Conference, 2004. EPTC 2004. Proceedinc
8-10 Dec. 2004 Page(s):289 - 293

AbstractPlus | Full Text: PDF(613 KB) IEEE CNF
Rights and Permissions

7. **A planar triangular monopole antenna for UWB communication**
Lin, C.-C.; Kan, Y.-C.; Kuo, L.-C.; Chuang, H.-R.;
Microwave and Wireless Components Letters, IEEE [see also IEEE Microwave Wave Letters]
Volume 15, Issue 10, Oct. 2005 Page(s):624 - 626
Digital Object Identifier 10.1109/LMWC.2005.856694

AbstractPlus | Full Text: PDF(304 KB) IEEE JNL
Rights and Permissions

8. **Application of statistical tools and methods for high density substrate pr development [printed circuit board fabrication]**
Martin, L.J.; Frei, J.;
Electronic Components and Technology Conference, 2002. Proceedings. 52nc
28-31 May 2002 Page(s):690 - 699
Digital Object Identifier 10.1109/ECTC.2002.1008173

AbstractPlus | Full Text: PDF(1008 KB) IEEE CNF
Rights and Permissions

9. **Characterization of coined solder bumps on PCB pads**
Jae-Woong Nah; Paik, K.W.; Won-Hoe Kim; Ki-Rok Hur;
Electronic Components and Technology Conference, 2002. Proceedings. 52nc
28-31 May 2002 Page(s):154 - 160
Digital Object Identifier 10.1109/ECTC.2002.1008089

AbstractPlus | Full Text: PDF(822 KB) IEEE CNF
Rights and Permissions

10. **Fill pattern and particle distribution of underfill material**
Yue Huang; Bigio, D.; Pecht, M.G.;
Components and Packaging Technologies, IEEE Transactions on [see also Cc Packaging and Manufacturing Technology, Part A: Packaging Technologies, II on]
Volume 27, Issue 3, Sept. 2004 Page(s):493 - 498
Digital Object Identifier 10.1109/TCAPT.2004.831776

AbstractPlus | References | Full Text: PDF(1912 KB) IEEE JNL
Rights and Permissions

11. **An assessment of the applicability of embedded resistor trimming and re**
Sandborn, P.A.;
Electronics Packaging Manufacturing, IEEE Transactions on [see also Compo and Manufacturing Technology, Part C: Manufacturing, IEEE Transactions on]
Volume 28, Issue 2, April 2005 Page(s):176 - 186
Digital Object Identifier 10.1109/TEPM.2005.848473

AbstractPlus | References | Full Text: PDF(1104 KB) IEEE JNL
Rights and Permissions

12. **A comprehensive study in thermal dissipation assessment of MPM-FCBC**

Mingzong Wang; Jeng Yuan Lai; Yu-Po Wang; Hsiao, C.S.;
Electronics Packaging Technology, 2003 5th Conference (EPTC 2003)
10-12 Dec. 2003 Page(s):491 - 496

AbstractPlus | Full Text: PDF(387 KB) **IEEE CNF**
Rights and Permissions

13. **Measuring Gold Thickness on Printed Circuit Boards for Control of Prod Processes**
Cooke, A.;
Components, Hybrids, and Manufacturing Technology, IEEE Transactions on | Trans. on Components, Packaging, and Manufacturing Technology, Part A, B,
Volume 5, Issue 1, Mar 1982 Page(s):69 - 73

AbstractPlus | Full Text: PDF(728 KB) **IEEE JNL**
Rights and Permissions

14. **Hybrid finite-difference/finite-volume time-domain analysis for microwav circuits with curved PEC surfaces using a nonuniform rectangular grid**
Mingwu Yang; Yinchao Chen; Raj Mittra;
Microwave Theory and Techniques, IEEE Transactions on
Volume 48, Issue 6, June 2000 Page(s):969 - 975
Digital Object Identifier 10.1109/22.846728

AbstractPlus | References | Full Text: PDF(240 KB) **IEEE JNL**
Rights and Permissions

15. **High performance package designs for a 1 GHz microprocessor**
Hasan, A.; Sarangi, A.; Baldwin, C.S.; Sankman, R.L.; Taylor, G.F.;
Electronic Components and Technology Conference, 2000. 2000 Proceedings
21-24 May 2000 Page(s):1178 - 1184
Digital Object Identifier 10.1109/ECTC.2000.853323

AbstractPlus | Full Text: PDF(688 KB) **IEEE CNF**
Rights and Permissions

16. **A heterodyne-scan phased-array antenna**
Kim, M.; Hacker, J.B.; Sailer, A.L.; Hong, J.H.;
Microwave and Guided Wave Letters, IEEE [see also IEEE Microwave and Wi Components Letters]
Volume 9, Issue 12, Dec. 1999 Page(s):535 - 537
Digital Object Identifier 10.1109/75.819423

AbstractPlus | References | Full Text: PDF(116 KB) **IEEE JNL**
Rights and Permissions

17. **A planar triangular monopole antenna for UWB communication**
Lin, C.-C.; Kuo, L.-C.; Chuang, H.-R.; Kan, Y.-C.;
Antennas and Propagation Society International Symposium, 2005 IEEE
Volume 3A, 3-8 July 2005 Page(s):512 - 515 vol. 3A
Digital Object Identifier 10.1109/APS.2005.1552300

AbstractPlus | Full Text: PDF(1096 KB) **IEEE CNF**
Rights and Permissions

18. **Investigation of printed wiring board testing by using planar coil type EC**
Yamada, S.; Fujiki, H.; Iwahara, M.; Mukhopadhyay, S.C.; Dawson, F.P.;
Magnetics, IEEE Transactions on
Volume 33, Issue 5, Part 1, Sept. 1997 Page(s):3376 - 3378
Digital Object Identifier 10.1109/20.617949

AbstractPlus | References | Full Text: PDF(304 KB) **IEEE JNL**
Rights and Permissions